

# **ANSYS Connection Users Guide**

**ANSYS Release 9.0**

002114  
November 2004

ANSYS, Inc. is a  
UL registered  
ISO 9001: 2000  
Company.



# **ANSYS Connection Users Guide**

**ANSYS Release 9.0**

ANSYS, Inc.  
Southpointe  
275 Technology Drive  
Canonsburg, PA 15317  
ansysinfo@ansys.com  
<http://www.ansys.com>  
(T) 724-746-3304  
(F) 724-514-9494

---

## Copyright and Trademark Information

Copyright © 2004 SAS IP, Inc. All rights reserved. Unauthorized use, distribution or duplication is prohibited.

ANSYS, DesignSpace, CFX, DesignModeler, DesignXplorer, ANSYS Workbench environment, AI\*Environment, CADOE and any and all ANSYS, Inc. product names referenced on any media, manual or the like, are registered trademarks or trademarks of subsidiaries of ANSYS, Inc. located in the United States or other countries. ICEM CFD is a trademark licensed by ANSYS, Inc. All other trademarks and registered trademarks are property of their respective owners.

ANSYS, Inc. is a UL registered ISO 9001: 2000 Company.

ANSYS Inc. products may contain U.S. Patent No. 6,055,541.

Microsoft, Windows, Windows 2000 and Windows XP are registered trademarks of Microsoft Corporation.

Inventor and Mechanical Desktop are registered trademarks of Autodesk, Inc.

SolidWorks is a registered trademark of SolidWorks Corporation.

Pro/ENGINEER is a registered trademark of Parametric Technology Corporation.

Unigraphics, Solid Edge and Parasolid are registered trademarks of Electronic Data Systems Corporation (EDS).

ACIS and ACIS Geometric Modeler are registered trademarks of Spatial Technology, Inc.

FLEXIm License Manager is a trademark of Macrovision Corporation.

This ANSYS, Inc. software product and program documentation is ANSYS Confidential Information and are furnished by ANSYS, Inc. under an ANSYS software license agreement that contains provisions concerning non-disclosure, copying, length and nature of use, warranties, disclaimers and remedies, and other provisions. The Program and Documentation may be used or copied only in accordance with the terms of that license agreement.

See the ANSYS, Inc. online documentation or the ANSYS, Inc. documentation CD for the complete Legal Notice.

If this is a copy of a document published by and reproduced with the permission of ANSYS, Inc., it might not reflect the organization or physical appearance of the original. ANSYS, Inc. is not liable for any errors or omissions introduced by the copying process. Such errors are the responsibility of the party providing the copy.

---

# Table of Contents

<b>1. Introduction to ANSYS Connection Products</b> .....	1-1
1.1. File Import Overview .....	1-1
1.2. Software Requirements .....	1-2
1.3. Importing Multiple Files (Create ANSYS Database) .....	1-2
1.4. Getting Started: Import a CAD File .....	1-3
<b>2. Importing Parts and Models</b> .....	2-1
2.1. ANSYS Connection for CATIA V4 .....	2-1
2.2. ANSYS Connection for CATIA V5 .....	2-2
2.3. ANSYS Connection for CIF .....	2-4
2.4. ANSYS Connection for Parasolid .....	2-5
2.4.1. Importing a Parasolid File .....	2-5
2.4.2. Parasolid Specific Attributes .....	2-6
2.5. ANSYS Connection for Pro/ENGINEER .....	2-7
2.6. ANSYS Connection for SAT .....	2-8
2.7. ANSYS Connection for Unigraphics .....	2-10
2.7.1. Additional Output Files from Unigraphics Conversions .....	2-11
2.7.2. Environment Variables for Unigraphics .....	2-12
<b>3. Starting ANSYS from Pro/ENGINEER and Unigraphics</b> .....	3-1
3.1. Launching ANSYS Connection from Pro/ENGINEER .....	3-1
3.1.1. UNIX System Configuration of Connection for Pro/ENGINEER .....	3-1
3.1.2. Windows System Configuration of Connection for Pro/ENGINEER .....	3-2
3.1.3. Launching ANSYS from Pro/ENGINEER .....	3-2
3.1.4. Modifying ANSYS Settings from Pro/ENGINEER .....	3-3
3.1.4.1. The ANSYS Configuration Settings .....	3-4
3.1.4.2. Search Rules for the config.anscon Configuration File .....	3-4
3.1.5. Troubleshooting Connection Problems .....	3-4
3.2. Launching ANSYS Connection from Unigraphics (UNIX Only) .....	3-4
3.2.1. Configuring Unigraphics .....	3-4
3.2.1.1. Running Unigraphics .....	3-5
3.2.2. Modifying ANSYS Settings from Unigraphics .....	3-6
3.2.2.1. Using the ANSYS Configuration Editor .....	3-7
3.2.2.1.1. Modifying an ANSYS Setting .....	3-7
<b>4. Importing Tutorial</b> .....	4-1
4.1. Importing a Unigraphics Part .....	4-1
A. Setting ANSYS Configuration Parameters .....	A-1
B. Troubleshooting ANSYS Connection Issues .....	B-1
B.1. General Troubleshooting .....	B-1
B.2. License Troubleshooting .....	B-1
B.3. Shared Library Problems .....	B-1
B.4. Configuration Errors for Pro/ENGINEER and Unigraphics Users .....	B-2
B.5. Troubleshooting Part Import .....	B-2
B.5.1. ANSYS Connection for Parasolid .....	B-3
B.5.2. Problems Specific to ANSYS Connection for Pro/ENGINEER .....	B-3
B.5.3. Problems Specific to ANSYS Connection for SAT .....	B-4
B.5.4. Problems Specific to ANSYS Connection for Unigraphics .....	B-5
Index .....	Index-1

## List of Figures

1.1. File Import Menus .....	1-3
2.1. ANSYS Connection for CATIA Dialog .....	2-2
2.2. ANSYS Connection for CATIA Version 5 Dialog .....	2-3
2.3. CIF Dialog Box .....	2-4
2.4. Connection for Parasolid Dialog .....	2-5
2.5. ANSYS Connection for PRO/E Dialog .....	2-7
2.6. Connection for SAT Dialog .....	2-9
2.7. ANSYS Connection for UG Dialog .....	2-10
3.1. The ANSYS 9.0 Menu .....	3-2
3.2. The Unigraphics Ansys Pull Down Menu .....	3-5
3.3. The ANSYS Configuration Editor .....	3-7
3.4. The Change Key Value Pop-Up Window .....	3-8
4.1. A Hinge Created in Unigraphics .....	4-1
4.2. The ANSYS Import Menu .....	4-2
4.3. The Connection for UG Dialog Box .....	4-2
4.4. The Unigraphics Hinge Displayed as an ANSYS Plot .....	4-3

## List of Tables

1.1. CAD Package and Recommended Connection Product .....	1-1
3.1. ANSYS Default Configuration Settings .....	3-3
3.2. ANSYS Configuration Settings .....	3-6

# Chapter 1: Introduction to ANSYS Connection Products

---

## 1.1. File Import Overview

The ANSYS Connection products provide you with a means to import CAD geometries into ANSYS. ANSYS Connection products can import geometries for the CAD/CAE file formats listed below:

- CATIA V5 R2-13
- CATIA V4.x or lower (separate Connection products must be used for CATIA V5 and CATIA V4)
- Parasolid 16 or lower
- Pro/ENGINEER 2001 and Wildfire 1/2
- SAT/ACIS 13 or lower
- Unigraphics NX 2.0
- Caltech Intermediate Format (CIF)

*Note* — ANSYS Connection products generally support prior releases of CAD files (such as ACIS 12.0); however, due to changes in the CATIA product, support is not available between CATIA 5.x and prior CATIA releases.

Table 1.1 provides a list of CAD packages, their file type, and the recommended ANSYS Connection product to use when importing files.

**Table 1.1 CAD Package and Recommended Connection Product**

CAD Package	File Type	Preferred ANSYS Connection Connection Product
CATIA 4.x and lower	<b>.model</b> or <b>.dlv</b>	CATIA
CATIA 5.x	<b>.CATPart</b> or <b>.CATProduct</b>	CATIA Version 5
Pro/ENGINEER	<b>.prt</b>	Pro/ENGINEER
Unigraphics	<b>.prt</b>	Unigraphics
Parasolid	<b>.x_t</b> or <b>.xmt_txt</b>	Parasolid
Solid Edge	<b>.x_t</b> or <b>.xmt_txt</b>	Parasolid
SolidWorks	<b>.x_t</b>	Parasolid
Unigraphics	<b>.x_t</b> or <b>.xmt_txt</b>	Parasolid
AutoCAD	<b>.sat</b>	SAT
Mechanical Desktop	<b>.sat</b>	SAT
SAT	<b>.sat</b>	SAT
Solid Designer	<b>.sat</b>	SAT

## Auto Launch Feature

ANSYS Connection allows you launch ANSYS from the Pro/ENGINEER or Unigraphics (UNIX Only) packages. This direct launch lets you to take advantage of ANSYS analysis features as you design with CAD software, thus augmenting design and testing processes.

*Note* — You must be working in a UNIX environment to launch ANSYS directly from Unigraphics. ANSYS will not automatically launch from any Windows platform.

## 1.2. Software Requirements

### OS and Hardware Requirements

Please refer to the appropriate ANSYS section for specific support information and system requirements.

### Software Installation Prerequisites

Make sure that your system conforms to the following requirements:

- Prior to initiating any ANSYS Connection product, ANSYS 9.0 must be installed and configured on the target system.
- As applicable, Pro/ENGINEER must be installed on your machine for proper ANSYS Connection operation.
- As applicable, Unigraphics must be installed on your machine for proper ANSYS Connection operation.
- A CAD file's geometry should be verified prior to initiating ANSYS Connection.
- For SAT, Parasolid, and CATIA, ANSYS Connection does not require any additional CAD product licenses.

## 1.3. Importing Multiple Files (Create ANSYS Database)

By default, ANSYS Connection writes ANSYS Neutral Format (**.anf**) files whenever a file is imported. A benefit of this feature is that an assembly (multiple **.anf** files) may be imported to create an ANSYS database by properly naming and saving individual files. The commands to perform this action are shown below.

**Command(s): /INPUT**

**GUI: File> Read Input From...**

*Note* — You cannot enter multiple files with an individual command; you must use a separate command line for each file. For example:

```
/INPUT part1.anf
/INPUT part2.anf
/INPUT part3.anf
```

ANSYS displays the following message after you import the second file (and any subsequent files):

```
You have already entered the input file utility (AUX15)
```

You can ignore this message.

The number of files that you can add to create a single database depends on the memory configuration of your machine.

ANSYS automatically increments the entity numbers of the objects (keypoints, lines, areas, and volumes) in the subsequent **.anf** files by the highest object in the previous **.anf** files. This prevents object numbers from being duplicated in the final database.

For example, you could have numbered lines and areas in **part1.anf** and **part2.anf** like this:

<b>part1.anf</b>	<b>part2.anf</b>	<b>part1.anf</b>	<b>part2.anf</b>
<b>Lines</b>	<b>Lines</b>	<b>Areas</b>	<b>Areas</b>
1, 2, 3, 4, 5, 6	1, 2, 3, 4, 5, 6	1, 2, 3, 4	1, 2



When you put both into a new ANSYS database, the object numbers are automatically updated like this:

Lines	Areas
1, 2, 3, 4, 5, 6, 7, 8, 9, 10, 11, 12	1, 2, 3, 4, 5, 6

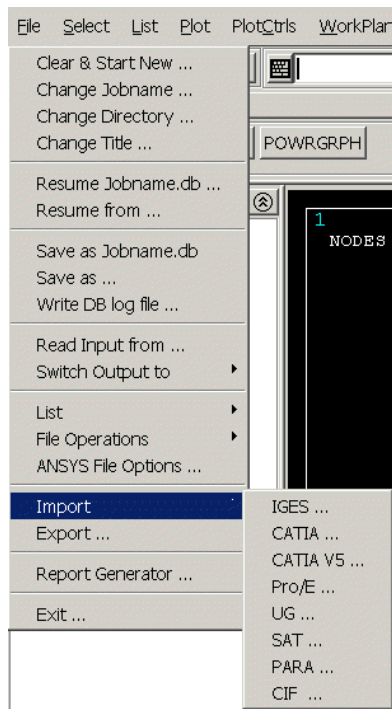
This combination scheme can produce some object number gaps in the new database.

## 1.4. Getting Started: Import a CAD File

This section examines how to open a CAD file with an ANSYS Connection product from the ANSYS graphical user interface (GUI). If you require additional information about navigating within the GUI, refer to Using the ANSYS GUI in the *ANSYS Operations Guide*.

1. Be sure that the following are running:
  - Current ANSYS version (ANSYS 9.0).
  - At least one ANSYS Connection product. If you have any questions about Connection's availability on your system, talk to your system administrator or refer to the appropriate installation guide.
2. Start ANSYS.
3. Select **File> Import** to display the menus shown in Figure 1–1.

**Figure 1.1 File Import Menu**



4. Select the applicable Connection product in the Import menu.

Import procedures specific to a CAD package are covered in the Importing Parts and Models section of this document.



# Chapter 2: Importing Parts and Models

---

This section further defines the steps to import parts or assemblies with ANSYS Connection products. Select the appropriate ANSYS Connection product to examine the import procedure for:

- CATIA 4.x and lower
- CATIA Version 5
- Caltech Intermediate Format (CIF)
- Parasolid
- Pro/ENGINEER. To start ANSYS from within Pro/ENGINEER, see Section 3.1: Launching ANSYS Connection from Pro/ENGINEER.
- SAT
- Unigraphics. To launch ANSYS from within Unigraphics, see Section 3.2: Launching ANSYS Connection from Unigraphics (UNIX Only). For a tutorial on importing a Unigraphics part, see Section 4.1: Importing a Unigraphics Part.

For troubleshooting information, see Appendix B; Troubleshooting ANSYS Connection Issues.

## 2.1. ANSYS Connection for CATIA V4

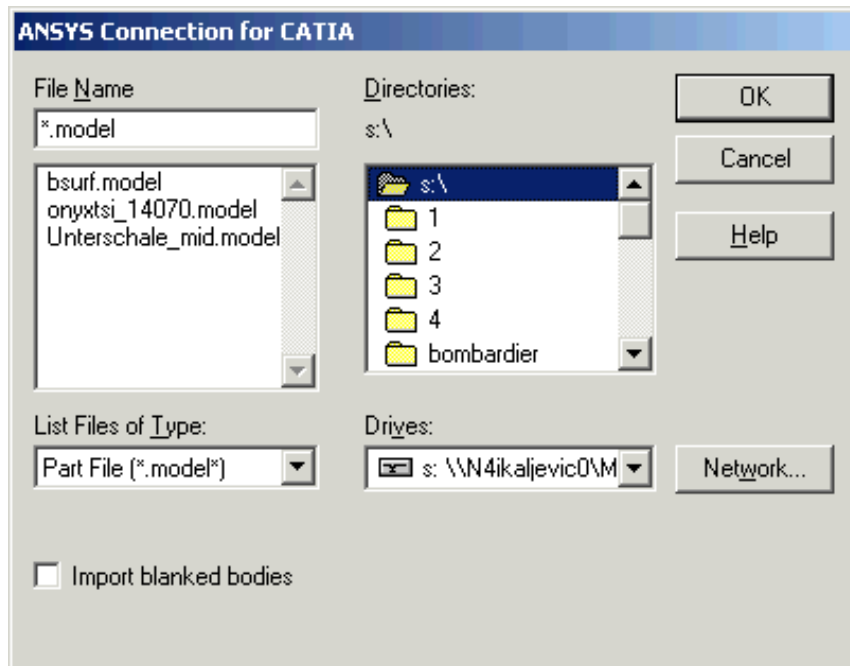
ANSYS Connection for CATIA supports models created with CATIA Version 4.x or lower. If you are using CATIA Version 5, see ANSYS Connection for CATIA V5.

Before you start ANSYS, the file (**.model** or **.dlv**) you wish to import must exist on your machine or on a known network location.

You can use the ANSYS GUI or the **~CATIAIN** command to initiate the import.

### Import with the ANSYS GUI

1. Select **File> Import> CATIA** from the menus. The **ANSYS Connection for CATIA** dialog box appears, as shown in the figure below.

**Figure 2.1 ANSYS Connection for CATIA Dialog**

2. Choose the appropriate file. If necessary, select the **Import blanked bodies** check box to allow suppressed CATIA data to be imported.
3. Click **OK**.

Once the geometry is imported, follow standard ANSYS analysis procedures; the *ANSYS Basic Analysis Guide* has many examples. If your imported model contains multiple volumes, Boolean operators are available to further process your geometry (refer to Section 5.4: Sculpting Your Model with Boolean Operations the *ANSYS Modeling and Meshing Guide* for more information).

*Note* — Only surface and volume models are imported. Wireframe models are not supported and will be filtered out.

## Import with the ANSYS ~CATIAIN Command

You can also use the following command line method to import a CATIA part.

Command Syntax:

```
~CATIAIN,Name,Extension,Path,-,-,BLANK,-,-
```

## 2.2. ANSYS Connection for CATIA V5

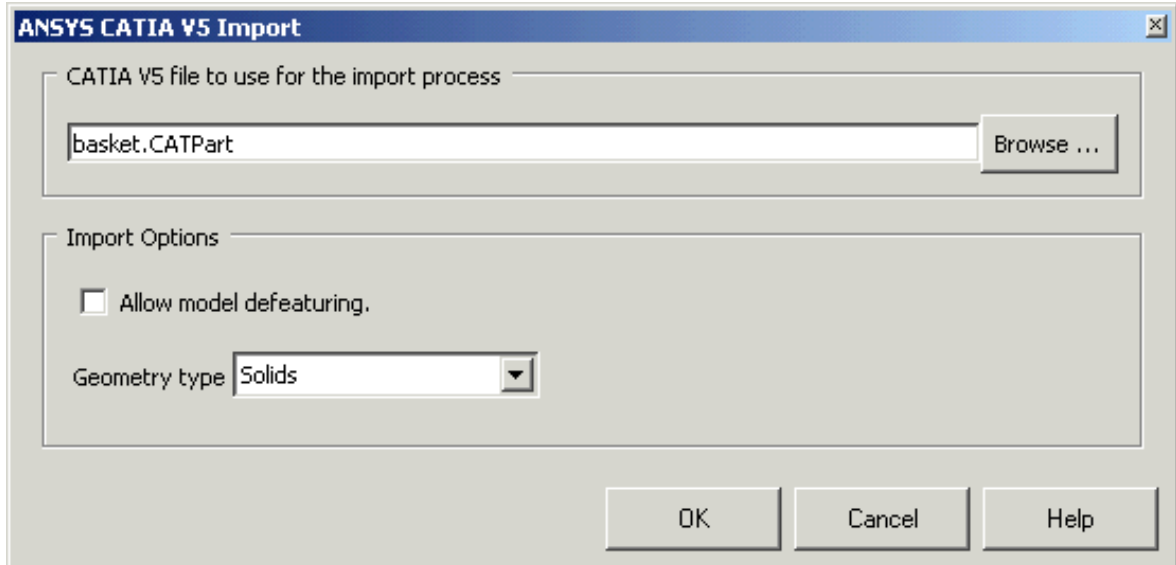
ANSYS Connection for CATIA Version 5 can import model part or assembly files created with CATIA V5 R2-13. Supported geometry files have the **.CATPart** or **.CATProduct** file extension.

Before starting ANSYS, the file (**.CATPart** or **.CATProduct**) you wish to import must exist on your machine or on a known network location. You can use the ANSYS GUI or the **~CAT5IN** command to import the file.

## Import with the ANSYS GUI

1. Select **File> Import> CATIA 5**. The ANSYS Connection for **CATIA Version 5 Import** dialog box appears, as shown in the figure below.

**Figure 2.2 ANSYS Connection for CATIA Version 5 Dialog**



2. Click the **Browse** button and navigate to the appropriate directory location of the part.
3. Select the part file.
4. If necessary, modify the **Import Options**:
  - Select **Allow model defeaturing** option to store the model in the solid database format so that it can be defeatured following import. If this option is not checked, the model is stored in a neutral database format, that restricts defeaturing following import. This is the default setting.
  - Select the **Geometry type** drop-down menu to filter the type of geometry contained in the CATIA version 5 file. Selections include:
    - **Solids** - imported as ANSYS volumes (default).
    - **Surfaces** - Imported as ANSYS areas.
    - **Wireframe** - Imported as ANSYS lines.
    - **All** - Import all entities; use this option when the file contains multiple entity types.
5. Press **OK** once selections are performed.

*Note* — If your imported model contains multiple volumes, Boolean operators are available to further process your geometry (see Section 5.4: Sculpting Your Model with Boolean Operations in the *ANSYS Modeling and Meshing Guide* for more information).

## Import with ANSYS ~CAT5IN Command

You can also use the following command line method to import a CATIA version 5 file.

Command Syntax:

**~CAT5IN**,Name,Extension,Path,Entity,FMT,NOCL,NOAN

## 2.3. ANSYS Connection for CIF

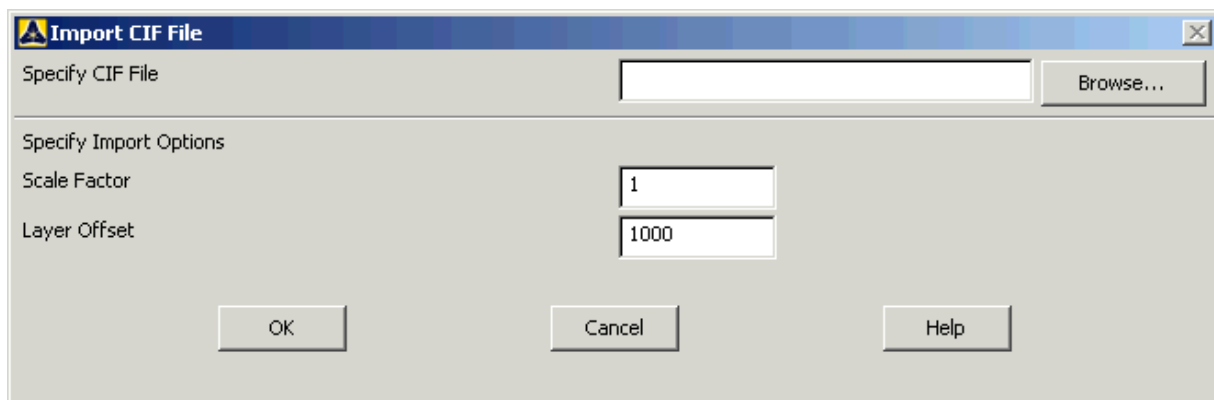
Caltech Intermediate Format (CIF) files are used in the electronics industry for the description of layered integrated circuit devices. The following procedure assumes that you are importing a CIF (.cif) geometry file.

You can import a CIF file using the ANSYS GUI or the **~CIFIN** command.

### Import with the ANSYS GUI

1. Select **File > Import > CIF**. The CIF dialog box displays, as shown in the figure below.

**Figure 2.3 CIF Dialog Box**



2. Select your CIF file by clicking the **Browse** button and navigating to the appropriate location or by entering a specific path in the **Specify CIF File** field. As necessary, modify the values of the following fields:
  - **Scale Factor** -- Enter the scale factor for the imported part or assembly. Default is 1.
  - **Layer Offset** -- Enter the layer offset. Default is 1000.

*Note* — Each layer has area, line, and keypoint components named A\_Layer, L\_Layer, and K\_Layer where “Layer” is the layer name on the CIF file. An assembly named “Layer” is created from all of the layer components. The material attribute number MAT (**MAT** command) is set to the layer number in the CIF file.

3. Click **OK**.

### Import with the ANSYS ~CIFIN Command

You can also use the following command line method to import your CIF file.

Command Syntax:

**~CIFIN**,Filename,Ext,Dir,Scale,Offset

## 2.4. ANSYS Connection for Parasolid

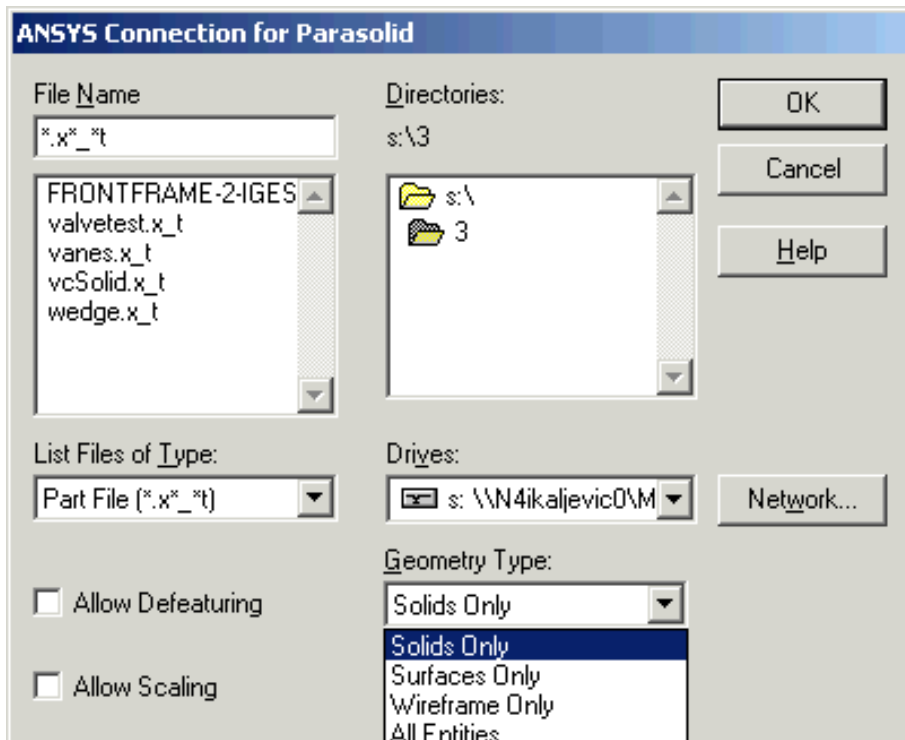
### 2.4.1. Importing a Parasolid File

Before you start ANSYS, the Parasolid file (.x\_t or .xmt\_txt) you wish to import must exist on your machine or on a known network location. You can use the ANSYS GUI or the **~PARAIN** command to import the file.

#### Import with the ANSYS GUI

1. Select **File > Import > PARA**. The **ANSYS Connection for Parasolid** dialog box appears, as shown in the figure below.

**Figure 2.4 Connection for Parasolid Dialog**



2. Select the desired Parasolid file. In addition to the file name, file location and file type options, modify the following values as necessary.
  - **Allow Defeaturing** - Check this box to store the model in the solid database format so it can be defeatured after import. If this is not checked, the model is stored in neutral database format, which restricts defeaturing after import (default).
  - **Allow Scaling** - Check this box to permit scaling of the model (if warranted by the model size). Please see the
  - **Geometry Type** - Import the file as:
    - **Solids Only** - Imported as ANSYS volumes (default).
    - **Surfaces Only** - Imported as ANSYS areas.
    - **Wireframe Only** - Imported as ANSYS lines.
    - **All Entities** - Import all entities; use this option when the file contains multiple entity types.

3. Click **OK** once selections are performed.

*Note* — If your imported model contains multiple volumes, Boolean operators are available to further process your geometry (see Section 5.4: Sculpting Your Model with Boolean Operations in the *ANSYS Modeling and Meshing Guide* for more information).

## Import with the ANSYS ~PARAIN Command

You can also use the following command line method to import your Parasolid file.

Command(s) Syntax:

`~PARAIN,Name,Extension,Path,Entity,FMT,Scale`

## Troubleshooting

For general troubleshooting information, please see Troubleshooting ANSYS Connection Issues, or examine Problems Specific to ANSYS Connection for Parasolid for specific issues. In addition, please refer to the Parasolid Specific Attributes section for additional information about specific considerations associated with importing Parasolid files.

### 2.4.2. Parasolid Specific Attributes

ANSYS Connection supports the import of multiple Parasolid files (**.anf**) for the creation of a single ANSYS database. You can import Parasolid files that make up an assembly, save them as **.anf** files then import them into one database. See Section 1.3: Importing Multiple Files (Create ANSYS Database) for more information on working with multiple **.anf** files.

However, models must be **saved** in the text Parasolid format. The saved model files must have either the **\*.x\_t** or **\*.xmt\_txt** extension.

## Scaling

When a Parasolid model is created, the measurement unit information is always represented by meters regardless of the units used to create the model. However, when the CAD package writes out the file, the measurement unit information is not included. The absolute numbers in the Parasolid file can differ from those represented in the ANSYS model.

When ANSYS reads a Parasolid file, the dimensions can be so small that ANSYS must apply a scale factor to improve the success of the transfer. ANSYS uses a scale factor of 10 or 100, depending on the original model size. ANSYS displays the scale factor used in the output window.

## Assemblies

You can directly import Parasolid assemblies into ANSYS using ANSYS Connection for Parasolid. After you import an assembly, you will have as many volumes in ANSYS as you had parts in the assembly. You must then do the following to build a contiguous model:

1. Glue volumes together using the **VGLUE** command.
2. Glue dissimilar meshes together by generating constraint equations at the interfaces using the **CEINTF** command. You should only use this method for models with small deflection.
3. Define bonded contact elements such as CONTA171 to glue dissimilar meshes together.



## 2.5. ANSYS Connection for Pro/ENGINEER

Before you start ANSYS, the Pro/ENGINEER part file (\*.prt\*) or assembly file (\*.asm\*) must exist on your machine or on a known network location. In addition, the file must have been created in Pro/ENGINEER 2001, Wildfire 1, or Wildfire 2 or an earlier version.

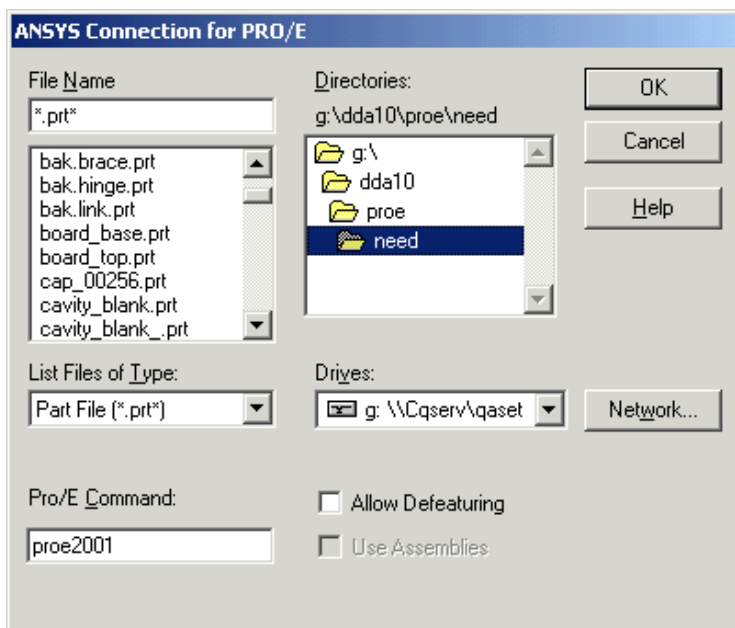
You can use the ANSYS GUI or the **~PROEIN** command to import the file.

*Note* — When importing Pro/ENGINEER parts and assemblies, ANSYS Connection for Pro/ENGINEER will always import the version with the highest number (**.asm.n**) extension, if a number is the final extension.

### Import with the ANSYS GUI

1. Select **File> Import> Pro/E**. The **ANSYS Connection for PRO/E** dialog box is displayed, as shown in the figure below.

**Figure 2.5 ANSYS Connection for PRO/E Dialog**



2. Select the desired Pro/ENGINEER file. As necessary, modify the values of the following dialog box options:
  - **Pro/E Command** - Specify the command that launches Pro/ENGINEER on your system. The default is **proewildfire**. This command cannot exceed 16 characters. The Pro/ENGINEER command name is set by the **ANSYS\_PROE\_CMD** environment variable.

*Note* — If you are a UNIX user, do not use an alias to run the Pro/ENGINEER executable. The path to the executable should be already set if the Pro/ENGINEER installation path has been defined for the **PATH** environment setting.

- **Allow Defeaturing** - Check this box to store the model in the solid database format so it can be de-featured after import. If this is not checked, the model is stored in neutral database format, which restricts defeaturing after import (default).
3. Click **OK**.

Remember that the geometry is always imported to ANSYS using the default ANSYS coordinate system (**CSYS,0**), Global Cartesian.

## Note for Windows Users

An additional window opens when you launch ANSYS from Pro/E. The window title bar contains the full path to the connection executable; the executable is named:

```
ac4pro90.exe
```

The message shown below will appear in the window.

```
Releasing Connection license...
```

In addition, you may see the following message at the start of geometry import on Windows systems:

```
INSTALL_SIGNAL_HANDLER: Failed to Install: Invalid argument
```

Ignore the above message and continue.

## Import with the ANSYS ~PROEIN Command

You can also use the following command line method to import your Pro/ENGINEER model.

Command Syntax:

```
~PROEIN,Name,Extension,Path,Proecomm,FMT
```

## Troubleshooting

For general troubleshooting information, please see Appendix B, Troubleshooting ANSYS Connection Issues, or Problems Specific to ANSYS Connection for Pro/ENGINEER for specific issues.

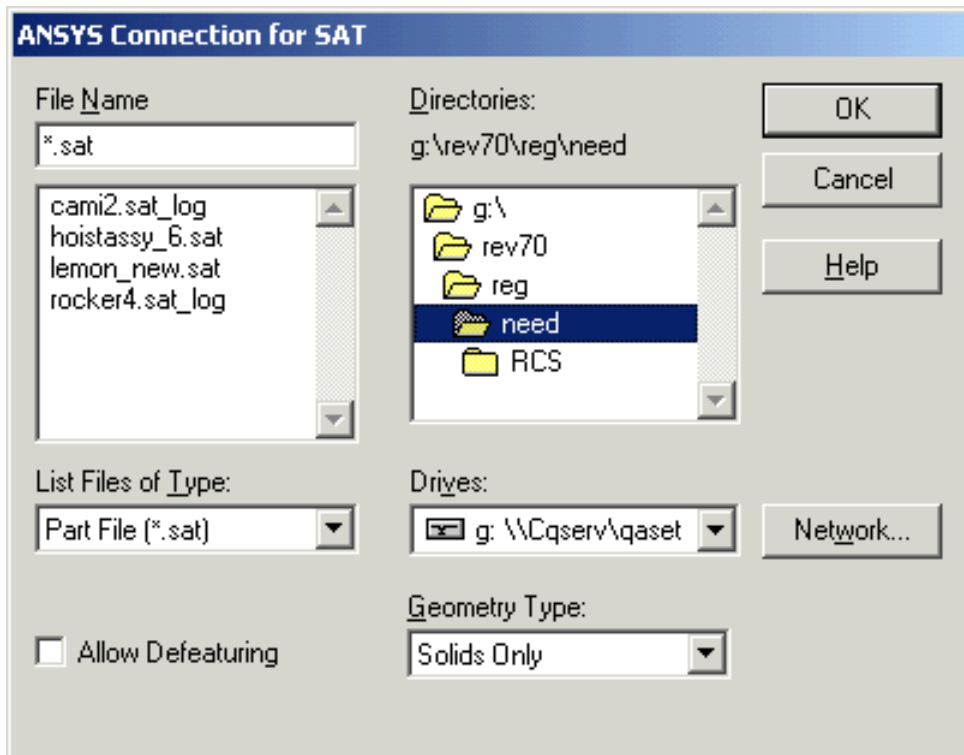
## 2.6. ANSYS Connection for SAT

ANSYS Connection products support the import of multiple **.anf** files for the creation of a single ANSYS database. You can import ACIS-based **.SAT** files that make up an assembly, save them as **.anf** files, and then compile them into one database. See Section 1.3: Importing Multiple Files (Create ANSYS Database) for more information on working with multiple **.anf** files.

You can import **.SAT** files using the GUI or by using the **~SATIN** command.

## Import using the ANSYS GUI

1. Select **File> Import> SAT**. The **ANSYS Connection for SAT** dialog box appears, as shown below.

**Figure 2.6 Connection for SAT Dialog**

2. Select the desired SAT file. In addition to the file name, location and type options, modify the following values as necessary.
  - **Allow Defeaturing** - Select this button to store the model in the solid database format so that it can be defeatured after import. The default is to store the model in neutral database format, which restricts defeaturing after import.
  - **Geometry Type** - Import the file as
    - **Solids Only** - imported as ANSYS volumes (default).
    - **SurfacesOnly** - Imported as ANSYS areas.
    - **Wireframe Only** - Imported as ANSYS lines.
    - **All** - Import all entities; use this when the file contains multiple entity types.
3. Click **OK**.

*Note* — If your imported model contains multiple volumes, Boolean operators are available to further process your geometry (see Section 5.4: Sculpting Your Model with Boolean Operations in the *ANSYS Modeling and Meshing Guide* for more information).

## Import using the ANSYS ~SATIN Command

You can also use the following command line method to import your SAT model.

Command Syntax

**~SATIN**,Name,Extension,Path,Entity,FMT,NOCL,NOAN

## Troubleshooting

For general troubleshooting information, please see [Troubleshooting ANSYS Connection Issues](#), or examine [Problems Specific to ANSYS Connection for SAT](#) for specific issues.

## 2.7. ANSYS Connection for Unigraphics

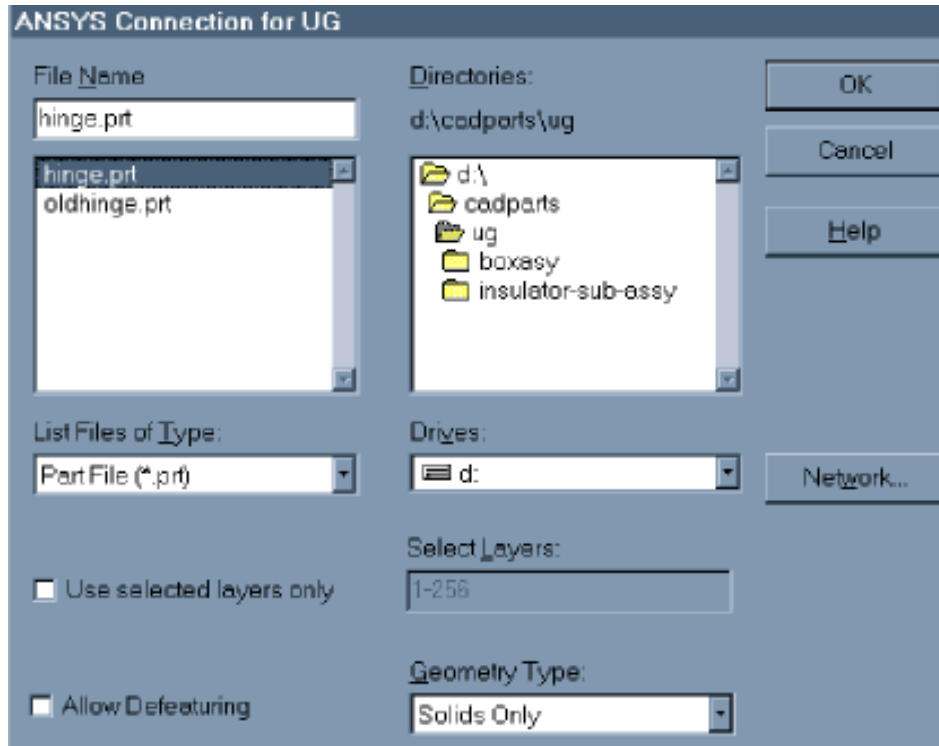
These procedures assume that you are running Unigraphics NX 2.0 and ANSYS 9.0. In addition, the part file you want to import **must** be created using Unigraphics.

When initiating a part file import, you can select the layers and entities to import, and can specify whether the part file is to be defeatured after import. You can use the ANSYS GUI or the **~UGIN** command.

### Import with ANSYS GUI

1. Select **File> Import> UG**. The **ANSYS Connection for UG** dialog box appears, as shown below.

**Figure 2.7 ANSYS Connection for UG Dialog**



2. Select the desired UG file. In addition to the file name, file location and file type options, you can modify the following values as necessary.
  - **Use selected layers only** - Import specific layers of the Unigraphics part by checking the **Use selected layers only** button and by entering the layer numbers to be imported in the field beside it. You can specify one layer (for example, 5) or a range of layers (for example, 7-20). The default is to import all layers (1-256).
  - **Allow Defeaturing** - Select this button to store the model in the solid database format so that it can be defeatured after import. The default is to store the model in neutral database format, which restricts defeaturing after import.

- **Geometry Type** - Import the part as:
  - **Solids Only** - imported as ANSYS volumes (default).
  - **Surfaces Only** - Imported as ANSYS areas.
  - **Wireframe Only** - Imported as ANSYS lines.
  - **All** - Import all entities; use this when the file contains multiple entity types.

3. Press **OK**.

## Import with ANSYS ~UGIN Command

You can also use the following command line method to import your Unigraphics model.

Command Syntax

```
~UGIN,Name,Extension,Path,Entity,LAYER,FMT
```

## Troubleshooting

For general troubleshooting information, please see [Troubleshooting ANSYS Connection Issues](#), or examine [Problems Specific to ANSYS Connection for Unigraphics](#) for specific issues.

### 2.7.1. Additional Output Files from Unigraphics Conversions

The Unigraphics-to-ANSYS import produces the following two files in addition to the database file.

- *partname.tbl*
- *partname.attr*

The *partname.tbl* file is the Unigraphics-to-ANSYS correspondence table file, which relates Unigraphics item names to ANSYS IDs and types.

Here is a sample Unigraphics-to-ANSYS correspondence table file:

```
!
! *** UNIGRAPHICS-TO-ANSYS CORRESPONDENCE TABLE FILE ***

! *** This version of the connection was validated with UG NX (V2.0.0.21)
!
!   Version      : 9.0 - UG NX V2.0
!   Build date   : 01/30/05
!   Generated on : 01/30/05 16:22
!   By           : rjg
!
!   Part         : hinge
!   Units        : Inches
!   Selected Layers : 1-256
!   Geometry Type : Solids Only
!
!   Translator Working Tolerance: 5.0000000E-04
!   Part system Tolerance (ref): 1.0000000E-03
!
!Object Name          ANSYS Type          ANSYS ID
TRM_SRF17            VOLUME              1
```

The Unigraphics attribute name appears in the first column, the ANSYS entity type appears in the second, and the ANSYS ID appears in the third.

The **partname.attr** file is the Unigraphics-to-ANSYS attribute file, which lists the object attributes that are applied in Unigraphics and that are for use in ANSYS. These attributes may be used to add mesh control information, load and boundary condition information, or material or element property information.

Here is a sample Unigraphics-to-ANSYS attribute file:

```
!
! *** UNIGRAPHICS-TO-ANSYS ATTRIBUTE FILE ***
!
! *** This version of the connection was validated with UG NX (V2.0.0.21)
!
! Version      : 9.0 - UG NX V2.0
! Build date   : 01/30/05
! Generated on: 01/30/05 16:22
! By          : rjg
!
! Part        : hinge
! Units       : Inches
! Selected Layers : 1-256
! Geometry Type : Solids Only
!
! Translator Working Tolerance: 5.0000000E-04
! Part system Tolerance (ref): 1.0000000E-03
!
!Object Name          Keyword          Value
```

## 2.7.2. Environment Variables for Unigraphics

Unigraphics and ANSYS Connection for Unigraphics require a number of environment variables to run properly. One of these variables is set by the ANSYS installation, and others are set by the Unigraphics installation. Because most of these environment variables are set during the installation, you will only need to reset them if you have used non-default installation directories.

### ANSCON\_CONFIG\_DIR

Sets the location of the **config.anscon** configuration file. If **ANSYS90\_DIR** has been set, you may not need to set this variable. For more information about the **config.anscon** file, please see Appendix A; Setting ANSYS Configuration Parameters

Default = **/ansys\_inc/ansys90/ac4/data**

*Note* — Both of the following variables must be set to import a part. Check to see what variables have been set for your platform, and if necessary, set these variables as indicated below.

### UGII\_BASE\_DIR

Sets the location of the Unigraphics files; set by the Unigraphics installation. A typical value for this variable is **/usr/ugs020**.

### UGII\_ROOT\_DIR

Sets the location of the Unigraphics executable; set by the Unigraphics installation. A typical value for this variable is **/usr/ugs020/bin**.

The next variable is not defined by the Unigraphics installation script. It can only be set in a **.cshrc** file or at command level before running Unigraphics:

### UGII\_VENDOR\_DIR

Defines the ANSYS menu for Unigraphics.

```
setenv UGII_VENDOR_DIR /ansys_inc/v90/ansys/ac4/bin/ug20/<platform>
```

If directory names on your platform contain uppercase characters, you must set the following environment variable:

**UGII\_OPTION**

Sets the case of characters in directory names on the current system. Possible options are MIXED, LOWER or UPPER. We recommend the following setting:

```
setenv UGII_OPTION MIXED
```





# Chapter 3: Starting ANSYS from Pro/ENGINEER and Unigraphics

---

ANSYS Connection allows you to launch ANSYS from within the Pro/ENGINEER and Unigraphics packages.

## 3.1. Launching ANSYS Connection from Pro/ENGINEER

This section examines the configuration requirements as well as the steps to launch ANSYS Connection from Pro/ENGINEER. The procedures outlined here assume that you have a working knowledge of Pro/ENGINEER and ANSYS, and that both of these products are installed on your system.

### 3.1.1. UNIX System Configuration of Connection for Pro/ENGINEER

1. Make sure that the **ANSYS90\_DIR** environment variable is set. If it is not set in your **.cshrc** or **.profile** file, set it from the UNIX command prompt. Please see the examples shown below. These examples assume that ANSYS 9.0 is installed in the default installation directory.

- For users running the C shell (csh), set the environment variable as:

```
setenv ANSYS90_DIR /ansys_inc/v90/ansys
```

- For users running the Korn (ksh) or Borne (sh) shell, set the environment variable as:

```
ANSYS90_DIR=/ansys_inc/v90/ansys  
export ANSYS90_DIR
```

2. If you are running any ANSYS product *other* than ANSYS Multiphysics, you must set the **ANSYS90\_PRODUCT** variable. Please refer to the Section 3.4: Product Variable Table in the *ANSYS, Inc. Licensing Guide* for a list of all available product variables. The following example illustrates how to set the variable if your default product is ANSYS Structural:

- For users running the C shell (csh), set the environment variable as:

```
setenv ANSYS90_PRODUCT STRUCT
```

- For users running the Korn (ksh) or Borne (sh) shell, set the environment variable as:

```
ANSYS90_PRODUCT=STRUCT  
export ANSYS90_PRODUCT
```

3. Verify that a copy of **config.anscon** is residing on the local machine. The default directory path for the file is:

**/ansys\_inc/v90/ansys/ac4/data/config.anscon**

If you wish to modify this file, first copy it to your working directory.

*Note* — The **ANSCON\_CONFIG\_DIR** environment variable is **not** used by ANSYS Connection for Pro/ENGINEER.

4. If you update Pro/ENGINEER after installing ANSYS Connection for Pro/ENGINEER, the process may remove the **protk.dat** file from the `/<ProE_install_dir>/<proe_platform>/text/usascii/` or the `<ProE_install_dir>/<proe_platform>/text` directory. If you launch ANSYS from Pro/ENGINEER, be sure that there is a copy of **protk.dat** in the **usascii** directory. If the **protk.dat** file does not exist, consult with your system administrator, or refer to the *ANSYS Installation and Configuration Guide for UNIX*.

### 3.1.2. Windows System Configuration of Connection for Pro/ENGINEER

1. Verify that a copy of **config.anscon** is residing on the local machine. The default folder path for the file is:

`drive:\Program Files\Ansys Inc\V90\ANSYS\ac4\data\<platform>\config.anscon`

If you wish to modify this file, first copy it to your working directory.

*Note* — The **ANSCON\_CONFIG\_DIR** environment variable is **not** used by ANSYS Connection for Pro/ENGINEER.

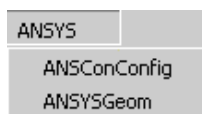
2. If you updated Pro/ENGINEER after installing ANSYS Connection for Pro/ENGINEER, the process may remove the **protk.dat** file from the `<proe_install>\<proe_platform>\text\<LANG>` or the `<proe_install>\<proe_platform>\text\` directories. If you launch ANSYS from Pro/ENGINEER, be sure that there is a copy of **protk.dat** in the **usascii** directory. If the **protk.dat** file does not exist, consult with your system administrator or refer to the *ANSYS Installation and Configuration Guide for Windows*.

### 3.1.3. Launching ANSYS from Pro/ENGINEER

The following procedure describes the steps to launch ANSYS from within the Pro/ENGINEER application. Please note that a part or assembly file **must** be open in Pro/ENGINEER before ANSYS can be launched.

1. Choose **File> Open** from the Pro/ENGINEER Menu Manager. The **File Open** dialog box appears.
2. Select the part you wish to open from the appropriate folder.
3. Click **Open**. The **part.prt** file opens in the top window.
4. Select **ANSYS Geom** from the ANSYS 9.0 menu, as shown in the figure below. The Pro/E part is converted into an ANSYS model database. If you encounter difficulties or the model does not display, please refer to the Troubleshooting an Imported File section listed below.

**Figure 3.1 The ANSYS 9.0 Menu**



*Note* — On Windows, you may see the following message: **INSTALL\_SIGNAL\_HANDLER: Failed to Install: Invalid argument**. If so, ignore the message and continue.

5. Analyze the part with ANSYS. If you have any questions about running an ANSYS analysis, see the *ANSYS Basic Analysis Guide*.
6. When the ANSYS analysis is complete, select **File> Quit**.

## Troubleshooting an Imported File

- If the model does not appear in the ANSYS window, select **Plot> Areas** or **Plot> Lines** to display the model.
- **Note for Windows Users:** An additional window may open when the connection to ANSYS is launched. The window title bar contains the full path to the connection executable; the executable is named **ac4pro.exe**. After the message **Releasing Connection license...** appears in the window, close the window. These messages are caused by Pro/ENGINEER and have no impact on your process.
- If the ANSYS options do not appear, the **protk.dat** file may not be properly installed. Based upon your platform, please refer to Section 3.1.1: UNIX System Configuration of Connection for Pro/ENGINEER or Section 3.1.2: Windows System Configuration of Connection for Pro/ENGINEER for additional information about installing the **protk.dat** file.

### 3.1.4. Modifying ANSYS Settings from Pro/ENGINEER

Before starting ANSYS from Pro/ENGINEER, you may wish to modify some ANSYS configuration settings. These configuration settings are stored in a text file called **config.anscon**, which you can copy into your working directory.

An example **config.anscon** file might look like this:

```
ANSYS_CMD C:\Program Files\ANSYS Inc\V90\ANSYS\bin\Intel\ansys.exe
ANSYS_GRAPHICS_DEVICE WIN32
ANSYS_SOLVER Frontal
ANSYS_SELECTED_LAYERS 1-256
ANSYS_GEOMETRY_TYPE Solids Only
ANSYS_NEUTRAL_FORMAT Yes
ANSYS_PRODUCT_NAME ANE3FL
```

The table shown below summarizes the default settings for the **config.anscon** file. For additional information, please refer to Appendix A; Setting ANSYS Configuration Parameters.

**Table 3.1 ANSYS Default Configuration Settings**

Keyword	Default Value	Description
<b>ANSYS_CMD</b>	Path name to the version of ANSYS to be run.	
	<b>UNIX:</b>  <b>/ansys_inc/V90/ansys/bin/ansys90</b> <b>Windows:</b>  <b>&lt;drive&gt;\program files\ansys inc\V90\ansys\bin\&lt;platform&gt;\ansys90.exe</b>	
<b>ANSYS_GRAPHIC_DEVICE</b>	<b>UNIX:</b> x11-stat <b>Windows:</b> win32	Graphics driver for ANSYS.[1]
<b>ANSYS_NEUTRAL_FORMAT</b>	YES	Store the part as a neutral file or as a solid model file.[2]
<b>ANSYS_PRODUCT_NAME</b>	ANSYS	The default ANSYS product, ANSYS Mechanical.[3]
<b>ANSYS_SOLVER</b>	Frontal	Included for compatibility only.[4]
<b>ANSYS_SELECTED_LAYERS</b>	1-256	
<b>ANSYS_GEOMETRY_TYPE</b>	Solids Only	

## Additional References

For additional information, please refer to the following:

1. *ANSYS Installation and Configuration Guide*.
2. Chapter 1, "Introduction to ANSYS Connection Products"
3. If you do not own ANSYS Mechanical, set this value to the default product for your machine. See the *ANSYS, Inc. Licensing Guide* for the complete list of ANSYS product names.
4. This setting is included only to maintain compatibility with the **config.anscon** file for ANSYS Connection for Unigraphics. It is not supported by Pro/ENGINEER and it is ignored by ANSYS.

### 3.1.4.1. The ANSYS Configuration Settings

### 3.1.4.2. Search Rules for the config.anscon Configuration File

The file containing the configuration settings may reside in a number of different directories. ANSYS Connection for Pro/ENGINEER searches for **config.anscon** in the following order:

1. Current working directory.
2. Home directory.
3. ../ansys90/ac4/data (if ANSYS 9.0 was installed in any location other than its default, the **/ansys90** portion of the path should be changed to that location).

When you save a new version of the **config.anscon** file, the new version is always saved to your working directory.

### 3.1.5. Troubleshooting Connection Problems

The problem scenario and solution discussed below is one that you may encounter when running ANSYS Connection for Pro/ENGINEER. Additional troubleshooting information may be found in Appendix B; Troubleshooting ANSYS Connection Issues.

ANSYS does not load when you select the **ANSYS\_GEOM** button.

This generally happens if you forgot to set the **ANSYS90\_DIR** environment variable as mentioned earlier in the Launching ANSYS Connection from Pro/ENGINEER section. Reset this variable and try again.

## 3.2. Launching ANSYS Connection from Unigraphics (UNIX Only)

This section examines the configuration requirements as well as the steps to launch ANSYS Connection from Unigraphics. The procedures outlined here assume that you have a working knowledge of Unigraphics and ANSYS, and that both of these products are installed on your system.

*Note* — While you can import Unigraphics parts into ANSYS on a Windows workstation, you cannot launch ANSYS from Unigraphics on a Windows workstation. You can only launch ANSYS from Unigraphics on certain UNIX platforms (see Section 1.2: Software Requirements).

### 3.2.1. Configuring Unigraphics

Verify and complete the configuration actions listed below.

1. Set the **ANSYS90\_DIR** environment variable. If this variable is not set in the **.cshrc** or **.profile** file, then set it at the UNIX command prompt. Examine the examples shown below. All sample paths use the default installation location of ANSYS 9.0 (**/ansys\_inc/v90/ansys**).

- For users running the C shell (csh), set the environment variable as:

```
setenv ANSYS90_DIR /ansys_inc/v90/ansys
```

- For users running the Korn (ksh) or Borne (sh) shell, set the environment variable as:

```
ANSYS90_DIR=/ansys_inc/v90/ansys
export ANSYS90_DIR
```

2. Verify that **UGII\_VENDOR\_DIR** is in your **.cshrc** file. This variable defines the ANSYS menu for Unigraphics. Add the following line to your **.cshrc**:

```
setenv UGII_VENDOR_DIR /ansys_inc/v90/ansys/ac4/bin/ug20/platform
```

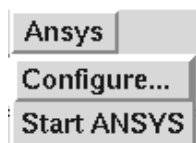
3. Verify that a copy of **config.anscon** is located on your machine. The default path name to this file on UNIX systems is **/ansys\_inc/v90/ansys/ac4/data/config.anscon**.

You can also use the **ANSCON\_CONFIG\_DIR** environment variable to define the location of the **config.anscon** file for a user or for a set of users.

### 3.2.1.1. Running Unigraphics

1. From a running Unigraphics session, open a part file or create a new Unigraphics part.
2. Select **Applications> ANSYS** from the menu bar to add the **ANSYS** drop-down menu to the menu bar. The **ANSYS** menu consists of two options: **Configure...** and **Start Ansys**, as shown in the figure below. The **Configure...** option is described in Section 3.2.2: Modifying ANSYS Settings from Unigraphics. The **Start Ansys** option launches ANSYS.

**Figure 3.2 The Unigraphics Ansys Pull Down Menu**



3. Select **Start Ansys** from the menu. After you select **Start Ansys**, the Unigraphics window closes temporarily, and the geometry transfer begins. Once the transfer is complete, the ANSYS menus and Graphics window displays.
4. Select **Plot> Volumes**. ANSYS displays the part model in the Graphics window. (If the model does not contain volumes, use one of the other options from the **Plot** menu to display the model.)
5. Perform an ANSYS analysis. If you have any questions about running an ANSYS analysis, see the *ANSYS Basic Analysis Guide*.
6. When the ANSYS analysis is complete, select **File> Quit**.
7. The ANSYS program closes, and the Unigraphics window re-displays.

### 3.2.2. Modifying ANSYS Settings from Unigraphics

Before starting ANSYS, you can specify some ANSYS settings from Unigraphics. For example, you may need to increase the memory size in order to run a large model successfully. To change these settings, select **Ansys> Configure...** from the Unigraphics menu bar. This displays the **ANSYS Configuration** editor as shown in Figure 3.3: “The ANSYS Configuration Editor”. The keywords and their values that are shown in the figure are read from the **config.anscon** file, an ASCII text file.

Connection for Unigraphics searches for the **config.anscon** file in the following locations and in the following order:

1. A directory defined by the **ANSCON\_CONFIG\_DIR** environment variable.
2. Working directory.
3. Home directory.
4. The ANSYS 9.0 product directory as defined by the **ANSYS90\_DIR** environment variable.

ANSYS stops searching as soon as the **config.anscon** file is found. If the **config.anscon** file from the ANSYS product directory is used, the file is copied to the working directory when you press the **Save** button from the **ANSYS Configuration**.

The table shown below provides a list of the ANSYS configuration variables, their default values, and a brief description of each setting.

**Table 3.2 ANSYS Configuration Settings**

Keyword	Default Value	Description
<b>ANSYS_CMD</b>	<b>UNIX:/ansys_inc/v90/ansys/bin/ ansys90</b>	Path name to the version of ANSYS to be run.[1]
<b>ANSYS_GRAPHIC_DEVICE</b>	<b>UNIXx11-stat</b>	Graphics driver for ANSYS.[2]
<b>ANSYS_MEMORY_SIZE</b>	512	Amount of RAM, in megabytes, suggested to run ANSYS.[2]
<b>ANSYS_SOLVER</b>	Frontal	Solver ANSYS will use.[3]
<b>ANSYS_SELECTED_LAYERS</b>	1-256	Layer or range of layers to be imported into ANSYS.[4]
<b>ANSYS_ENTITY_TYPE</b>	Solids Only	Type of entities ANSYS will import.[4]
<b>ANSYS_NEUTRAL_FORMAT</b>	Yes	Store the part as a neutral file or as a solid model file.[4]
<b>ANSYS_PRODUCT_NAME</b>	ANSYS	The default ANSYS product, ANSYS Multiphysics.[5]

### Additional References

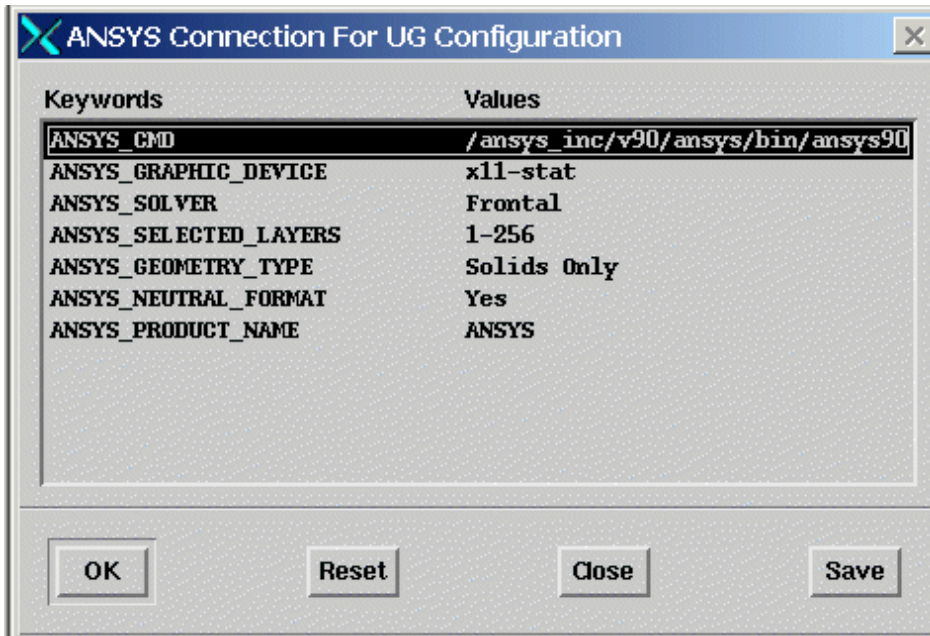
1. Pro/ENGINEER cannot process UNC paths. Always use drive letters rather than UNC paths in the Pro/ENGINEER configuration files.
2. See the appropriate *ANSYS Installation and Configuration Guide*.
3. See the *ANSYS Basic Analysis Guide*.
4. See **~UGIN**.
5. See Table 3–1 in Licensing for the complete list of ANSYS product names.

### 3.2.2.1. Using the ANSYS Configuration Editor

The Configuration Editor allows you to modify keyword values. To do so, follow the steps listed below.

1. Display the **ANSYS Connection For UG Configuration** window. This window is the configuration editor.

**Figure 3.3 The ANSYS Configuration Editor**



2. Selection the Keyword you wish to modify and then click **OK**. You may also select a Keyword by double-clicking its value in the list. The **Change Key Value** dialog box appears displaying the selected keyword. Depending on the keyword, the value may be changed through an edit field or an option button from this dialog box.
3. Enter a new value. It displays in the **ANSYS Configuration** dialog box.
4. Click **Save** from this dialog to save the modified value(s).

#### Notes

The **Reset** button resets the settings with the values read in from the configuration file.

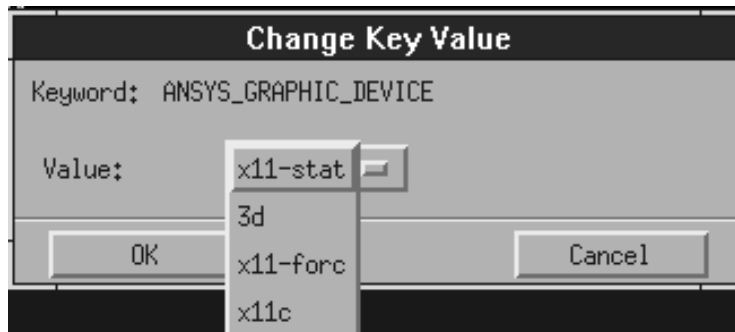
The **Close** button closes the **ANSYS Configuration** editor.

#### 3.2.2.1.1. Modifying an ANSYS Setting

The following procedure outlines the steps to modify an ANSYS setting using the **ANSYS Configuration Editor**. In this example, a graphic device is modified.

1. Select a keyword and click **OK** or double-click a keyword. A dialog box related to the keyword appears. In this case, the dialog box named **Change Key Value**.
2. Select the option button to display the available values for the **ANSYS\_GRAPHIC\_DEVICE** keyword. The figure shown below illustrates the values.

**Figure 3.4 The Change Key Value Pop-Up Window**



3. Select 3d and the click **OK**. The new value appears in the **ANSYS Configuration** dialog box.
4. Click **Save** to save your modification.
5. Click **Close** to exit the dialog box.

For general troubleshooting information, please refer to Appendix B:, Troubleshooting ANSYS Connection Issues.



# Chapter 4: Importing Tutorial

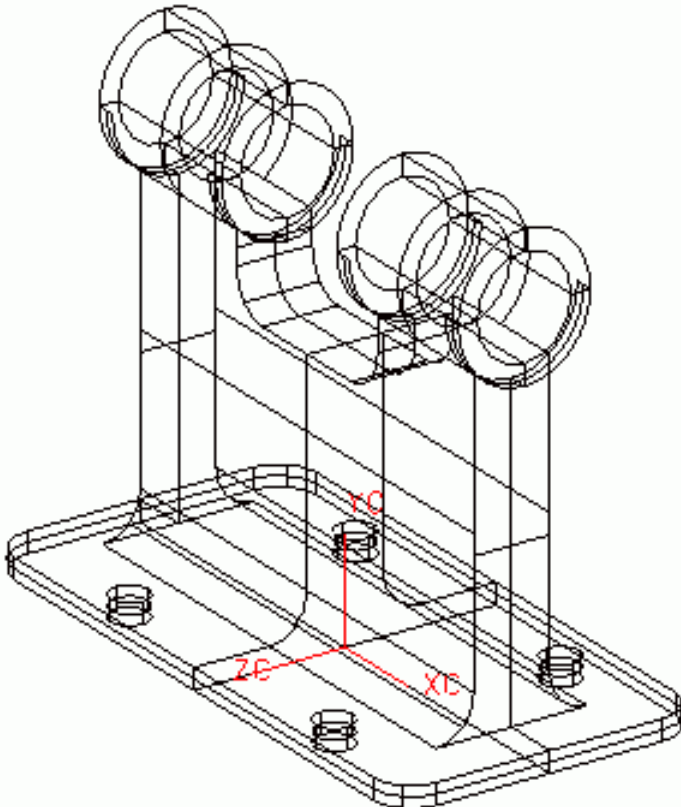
---

## 4.1. Importing a Unigraphics Part

This section provides a tutorial for importing parts from a CAD system into ANSYS. Because the procedure is similar in each case, we will use a Unigraphics part import as an example.

The part below is a hinge that was created using Unigraphics.

**Figure 4.1 A Hinge Created in Unigraphics**



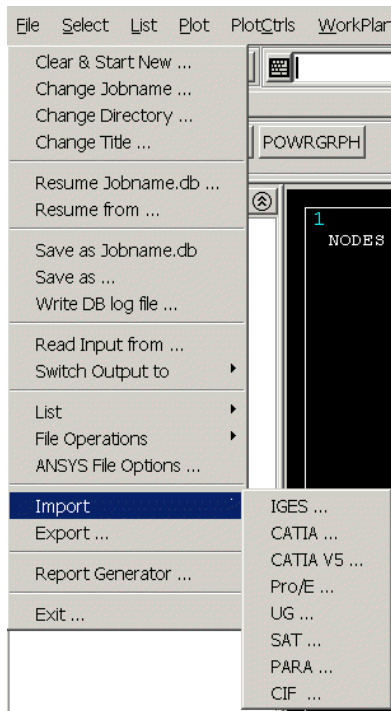
1. Start ANSYS in graphics mode. Type the following at the command level:

```
ansys90 -g
```

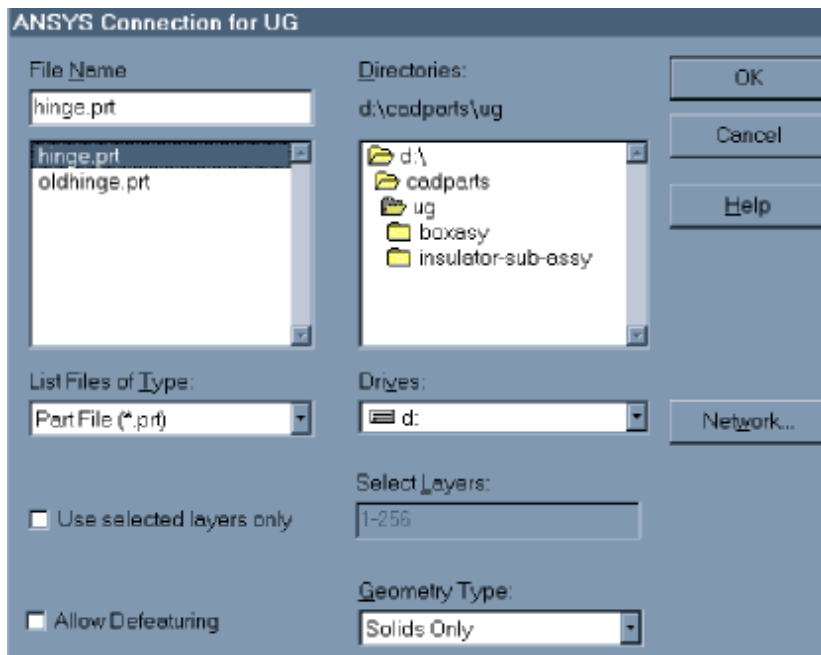
Before you import a file into ANSYS, it is important that you begin with an *empty* database. If you have already run an analysis, be sure to clear the database by invoking the **/CLEAR** command (**File> Clear & Start New**) before beginning the file import process.

*Do not neglect this step* - if the current database is not empty, the database may become corrupted once you import the Unigraphics part.

2. Select **File> Import> UG**.

**Figure 4.2 The ANSYS Import Menu**

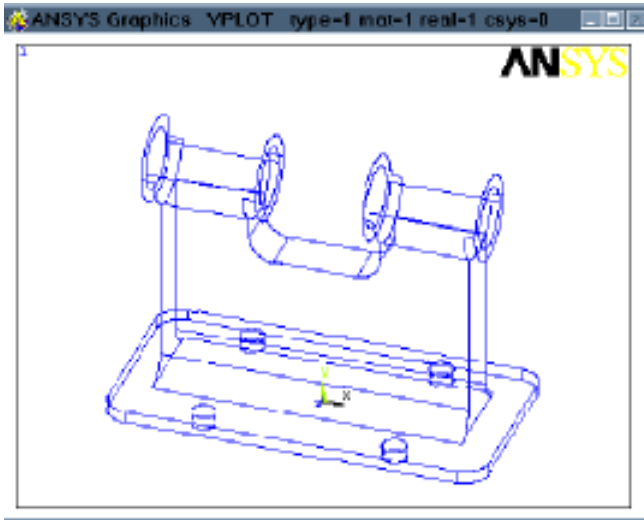
The following **ANSYS Connection for UG** dialog box is displayed.

**Figure 4.3 The Connection for UG Dialog Box**

3. Select the part file that you wish to import. Only Unigraphics parts can be transferred, and only one part may be selected at a time.
4. If you want to import a subset of layers, check the **Use Selected Layers Only** box and then enter the layer number(s) in the **Select Layers** field. You can specify a single layer (for example, 5) or a range of layers (for example, 5-20). If the button is not checked, all layers (1-256) are imported by default.

5. If you plan to defeature the part after import, select the **Allow Defeaturing** box. By default, the part is imported in a neutral database format that restricts defeaturing.
6. Select the **Geometry Type** option button to display a list of entities that could be imported: **Solids Only**, **Surfaces Only** (known as sheets in Unigraphics), **Wireframe Only**, or **All Entities**.
7. Click **OK** when your selections and settings are complete. ANSYS processes the file and displays a plot of the hinge, as illustrated in the figure below.

**Figure 4.4 The Unigraphics Hinge Displayed as an ANSYS Plot**



8. Perform the necessary steps to complete an ANSYS analysis. See the *ANSYS Basic Analysis Guide* if you have questions about an ANSYS analysis.



# Appendix A. Setting ANSYS Configuration Parameters

This appendix discusses the configuration of ANSYS using the **config.anscon** configuration file. This configuration file is used to configure ANSYS when it is launched from Pro/ENGINEER and Unigraphics. Default paths include:

## UNIX Systems

**/ansys\_inc/v90/ansys/ac4/data/config.anscon**

## Windows Systems

**drive:\Program Files\Ansys Inc\V90\ANSYS\ac4\data\<platform>\config.anscon**

The **config.anscon** file may be modified by the system administrator to set information for *all* ANSYS Connection users on a system. However, you can modify this file for your projects.

The following list defines the configuration settings that you can set in the **config.anscon** file, and the typically default settings.

### ANSYS\_CMD

Sets the full path name of the ANSYS software to be run. The UNIX default is **/ansys\_inc/v90/ansys/bin/ansys90**; the Windows default is **drive:\Program Files\Ansys Inc\V90\ANSYS\bin\ansys90**

### ANSYS\_GRAPHIC\_DEVICE

Sets the graphics driver to be used by ANSYS. The possible values for this keyword are x11-stat, 3d, x11-forc, and x11c. The UNIX default is x11. The Windows default is win32.

### ANSYS\_MEMORY\_SIZE

Sets the amount of ANSYS workspace, in megabytes, required to run ANSYS. The value you use is limited by how much swap space or virtual memory is available on your workstation. The default value is 512 MB.

### ANSYS\_PRODUCT\_NAME

Sets the product name for ANSYS. See Product Variable Table in the *ANSYS, Inc. Licensing Guide* for the complete list of ANSYS product names. The default is system dependent, but typically ANSYS (or ANSYS Multiphysics).

### ANSYS\_NEUTRAL\_FORMAT

Sets the database format in which the ANSYS model will be saved. The possible values are Yes (save as neutral format) or No (save as solid format). The default value is Yes (save as neutral).

### ANSYS\_SOLVER

Sets the ANSYS solver that you want to run to determine a model's solution. The possible values for this keyword are Frontal and Power. Refer to the Section 3.2: Selecting a Solver *ANSYS Basic Analysis Guide* for more information on solvers. This variable is only active for ANSYS Connection for Unigraphics. The default value is Frontal.

### ANSYS\_SELECTED\_LAYERS

Selects the layers of the part to transferred to ANSYS. One layer or a range of layers may be specified. This variable is only active for ANSYS Connection for Unigraphics. The default value is 1-256 (transfer all layers).

### ANSYS\_ENTITY\_TYPE

Sets the entities of the CAD part that will be transferred to ANSYS. The possible values are Solids Only (import as ANSYS volumes), Surfaces Only (import as ANSYS areas), Wireframe Only (import as ANSYS lines) or All (import all entities). This variable is only active for ANSYS Connection for Unigraphics. The default value is Solids Only.

While the following settings are included in the Connection for Pro/ENGINEER **config.anscon** file, those settings are ignored: **ANSYS\_SOLVER**, **ANSYS\_SELECTED\_LAYERS** and **ANSYS\_ENTITY\_TYPE**. The values for these keywords are only used by ANSYS Connection for Unigraphics.

In addition to supporting the **Configuration Editor**, ANSYS Connection for Unigraphics supports the **ANSCON\_CONFIG\_DIR** environment variable, which lets you define the location of the **config.anscon** file. ANSYS Connection for Pro/ENGINEER does *not* support this variable.

# Appendix B. Troubleshooting ANSYS Connection Issues

This appendix examines some common error messages that you may encounter. It also discusses general ANSYS Connection troubleshooting as well as licensing issues that may arise.

The following sections describe some of the primary problems you may encounter when importing parts into ANSYS. The General Troubleshooting section outlines problems common to any ANSYS Connection product. Later sections discuss general problems with part import. For information on problems specific to a particular ANSYS Connection product, please refer to the appropriate section of the user documentation.

## B.1. General Troubleshooting

### The software fails to run

Be certain the programs have execute permissions, and that there is adequate swap space to run ANSYS and the CAD package simultaneously. In addition, if you do not possess the proper licensing, the system may not run.

If you receive one of these error messages show below, check the ANSYS text window for additional error or warning messages. When an ANSYS Connection product is running, an overly-long command string can be passed to the CAD package. For example, on Silicon Graphics machines, the total length of a passed command string cannot exceed 75 characters. If this message occurs, copy the file to the current working directory and try to import it again.

#### Error Messages:

*filename.anf* not found

*filename.aaa* not found

## B.2. License Troubleshooting

ANSYS Connection will not run

Make sure that you have a current license to run the appropriate connection product. Check the license file in the licensing directory (**/ansys\_inc/shared\_files/licensing** (UNIX) or **<drive>:Program Files\Ansys Inc\Shared Files\licensing** (Windows)). See the *ANSYS, Inc. Licensing Guide* for a complete list of product names and their associated feature name.

### **This feature is not available or Connection for CAD\_package is not available**

These messages indicate that either the ANSYS Connection product is not properly installed or it has not been licensed for the current system. In addition, if you try to run ANSYS Connection for Pro/ENGINEER or Unigraphics you must also have a valid licenses for those products. Contact your system administrator if you have any questions about which products are licensed for your system.

## B.3. Shared Library Problems

An error was detected while operating on a shared library:

PATH=C:\Program Files\Ansys Inc\V90\ANSYS\lib\intel\libac4.so CMD=~SATIN FNC=AnsysConnectCad

This message indicates that ANSYS and ANSYS Connection were installed on a shared drive (named by using the Universal Naming Convention) rather than on a local drive (named by using a drive letter). Replace the drive letter in the configuration file with the UNC name of the drive. Refer to the *ANSYS Installation and Configuration Guide for Windows* for more information.

## B.4. Configuration Errors for Pro/ENGINEER and Unigraphics Users

\*\*\* FATAL \*\*\* The database (-db) space requested is not currently available. Reenter ANSYS command line with less database space requested.

Indicates part transfer failure in Unigraphics and Pro/ENGINEER if **ANSYS\_MEMORY\_SIZE** is set to a value *greater* than the available memory available on the machine. Use the **ANSYS Configuration** editor to reduce the amount of RAM set for the **ANSYS\_MEMORY\_SIZE** variable and relaunch ANSYS.

## B.5. Troubleshooting Part Import

This section examines some general troubleshooting techniques when importing a part.

- If you receive errors, examine the **filename.log** file created by ANSYS. It is located in the current working directory and may contain clues to the source of the error.
- If a part name fails to be read into ANSYS. Be sure that the part name begins with an alphanumeric (required). If your part name begins with a special character, you must rename it before importing. For example, a Unigraphics user can rename the part name **\_123hinge.prt** to be **123hinge.prt**, or **x123hinge.prt**.
- Models that have undergone multiple conversions from other CAD packages may cause difficulties when transferring into ANSYS.
- If your CAD package has a "defeature" option, use the utility *before* importing the part to ANSYS. This allows you to avoid analyzing a model with small features that are irrelevant in an ANSYS study. Parts that have sharp vertices or areas that have lines that touch may experience problems in tessellation and meshing.
- If the model fails to mesh, you can repair, simplify, or reimport the model.
  - Choose **Main Menu > Preprocessor > Modeling > Check Geom > Show Degeneracy** for a list of possible problem areas in your model. To use this commands, the part must be imported with the **Allow Defeathering** option selected.
  - Repair portions of the model by choosing **Main Menu > Preprocessor > Modeling > Update Geom** to update nodes. To use this command, the part must be imported with the **Allow Defeathering** option selected.
  - Be sure to use ANSYS elements that are designed to model irregular meshes. These elements are particularly well-suited for using with imported parts:

PLANE2	SOLID87	SOLID98	SOLID123
PLANE35	SOLID92	SOLID122	HYPER158

- If the mesh still fails, try the **ANFO** command. Enter the following commands in the ANSYS Input window:

```
/AUX15
ANFO, PROJ, ALTER
/INP, partfile, anf
```

Replace **partfile** with the name of a CAD part file. This sequence of commands restarts the analysis in the working directory and can only be used to create ANSYS neutral files.

**Caution:** Running **ANFO** can take a very long time. It can take hours to rerun even a very small model. Consider running this process overnight. **ANFO** will be able to import and create areas and volumes for some files that failed in earlier attempts.



**Cannot project lines onto surface**

Areas and volumes are not being properly imported into ANSYS. To fix this problem, clear the ANSYS database and run the **ANFO** command as described in the previous paragraph.

**B.5.1. ANSYS Connection for Parasolid**

The following issues may be encountered when running ANSYS Connection for Parasolid. For general troubleshooting information, see Appendix B; Troubleshooting ANSYS Connection Issues.

If a part fails to convert into ANSYS, examine **filename.para\_log** and **filename.ans\_log**. Both files are located in the working directory. The **filename.para\_log** file contains warnings and errors generated during the conversion that relate to problems in the file itself. The **filename.ans\_log** file contains warnings and errors that relate to building the model in ANSYS.

*Note* — The **filename.ans\_log** file is only generated when the part is saved in neutral database format.

**Failed to process.**

May indicate a corrupt Parasolid file. This can be caused by sending a Parasolid file via e-mail, or by trying to import a binary file. The safest way to transfer a Parasolid file is to use FTP and transfer the file using binary mode; be certain to save the file in ASCII format (not binary).

**Failed to process.**

Schema keys are missing. Check the installation. Schema keys should be found in **/ansys\_inc/v90/ansys/ac4/schema** (UNIX) or in **<drive>:\Program Files\Ansys Inc\V90\ANSYS\ac4\schema** (Windows). On Windows machines, the **P\_SCHEMA** environment variable must be set to **<drive>:\Program Files\Ansys Inc\V90\ANSYS\ac4\schema**.

**\*\*\*Warning: ANSYS environment variables: ANSYS90\_DIR and/or ANSYS90\_SYSDIR are not set.**

**\*\*\*ERROR: ANSYS Connection for Parasolid directory structure is not set. Unable to find the Connection executable.**

Either the **ANSYS90\_DIR** or the **ANSYS90\_SYSDIR** variable needs to be set. See the *ANSYS Installation and Configuration Guide for Windows* or the *ANSYS Installation and Configuration Guide for UNIX* for information about these configuration variables.

**B.5.2. Problems Specific to ANSYS Connection for Pro/ENGINEER**

Listed below are several problems you may see when running ANSYS Connection for Pro/ENGINEER. For general troubleshooting information, see Appendix B; Troubleshooting ANSYS Connection Issues.

Please note that it is necessary to first run the **ANS\_ADMIN utility** to configure the Connection for Pro/E. See the ANSYS Installation and Configuration Guide (for UNIX or Windows) for more information.



**ANSYS Connection for Pro/ENGINEER can't find necessary files to run.**

ANSYS expects Pro/ENGINEER to be run in the same directory from which it is started (the ANSYS working directory). The trouble can occur if you have a startup file (such as **prowildfire.bat**, **proe2001.bat**, etc.) that modifies the working directory of Pro/ENGINEER (typically from a change directory, `cd`, command).

**ANSYS Connection for Pro/ENGINEER does not run.**

Be sure Pro/ENGINEER can run on the machine on which you just installed ANSYS Connection for Pro/ENGINEER, and that it can be launched using the command displayed in Figure 2.5: "ANSYS Connection for PRO/E Dialog".

Verify that a **protk.dat** file exists in one of the directories documented in ANSYS Installation and Configuration Guide for your platform. Verify that the path names listed in this file are correct.

Verify that the **protk.dat** file is pointing to the correct executable: `ac4pro`, `ac4pro.wildfire`, or `ac4pro32`. See the appropriate *ANSYS Installation and Configuration Guide* for more information.

**B.5.3. Problems Specific to ANSYS Connection for SAT**

Listed below are several problems you may encounter when running ANSYS Connection for SAT. For further troubleshooting information, see Appendix B; Troubleshooting ANSYS Connection Issues.

If a part fails to convert into ANSYS, examine **filename.sat\_log** and **filename.ans\_log**. Both files are located in the working directory. The **filename.sat\_log** file contains warnings and errors generated during the conversion that relate to problems in the file itself. The **filename.ans\_log** file contains warnings and errors that relate to building the model in ANSYS.

If a model is sensitive to tolerance, shapes may not be created properly, which can lead to poorly transferred areas. If you find this problem with a model, reimport it using the following steps:

1. Import the file using the default format (neutral).
2. Enter the following command in the ANSYS Input window:

```
XANF,filename.aaa,-tolerance_1.0e-6
```

where **filename.aaa** is the name of the file you are working with

**The following problem may occur on UNIX systems:** The software may not come up if the required shared library (**ansys\_inc/ansys90/lib/<platform>/libac4.so**) is not present or if the path names listed in this file are incorrect. Check for proper installation (see *ANSYS Installation and Configuration Guide for UNIX*) and reinstall the software.

**The following problems may occur on Windows systems:** A sys error may be displayed if the executable is in the wrong directory or if the required shared libraries are missing. Check for proper installation (see *ANSYS Installation and Configuration Guide for Windows*) and reinstall the software.

**\*\*\*Warning: ANSYS environment variables: ANSYS90\_DIR and/or ANSYS90\_SYSDIR are not set.**

**\*\*\*ERROR: ANSYS Connection for SAT directory structure is not set. Unable to find the Connection executable.**

Either the **ANSYS90\_DIR** or the **ANSYS90\_SYSDIR** variable needs to be set. See the *ANSYS Installation and Configuration Guide for Windows* or the *ANSYS Installation and Configuration Guide for UNIX* for information about these configuration variables.

## B.5.4. Problems Specific to ANSYS Connection for Unigraphics

Listed below are several problems you may encounter when running ANSYS Connection for Unigraphics.

ERROR MESSAGE RECEIVED:

\*\*\* WARNING \*\*\*

File <filename>.anf does not exist.

The /INPUT command will be ignored.

The Unigraphics root directory (UGII\_ROOT\_DIR) must be appended to the system PATH. This must be done manually prior to running the Connection for Unigraphics product.



Example:

If the UGII\_ROOT\_DIR is:

**UGII\_ROOT\_DIR=C:\Program Files\EDS\Unigraphics NX 2.0\UGII\.**

Then the PATH should include:

**Path=C:\WINNT\system32;C:\WINNT;C:\WINNT\System32\Wbem;C:\Program Files\EDS\Unigraphics NX 2.0\UGII\;....**

If a part fails to open, and directory names in the path contain uppercase characters, see Section 2.7.2: Environment Variables for Unigraphics. You need to set the **UGII\_OPTION** environment variable to accommodate uppercase characters.

Before you import a part into ANSYS, you should use the Examine Geometry function in Unigraphics to identify any abnormalities in the part. Any problems with the part need to be corrected in Unigraphics first.

Parts created using an older version of Unigraphics (especially those created prior to version 11.0) may need to be upgraded to a supported version of Unigraphics.

If a part fails to import into ANSYS, examine **jobname.ug\_log** and **jobname.ans\_log**. Both files are located in the current working directory. The **jobname.ug\_log** file contains warnings and errors generated during the conversion that relate to problems in the part itself. The **jobname.ans\_log** file contains warnings and errors that relate to building the model in ANSYS.

When you import a part and only certain aspects of it are displayed (for example, the keypoints), look at the part in Unigraphics using the Examine Geometry tool. This tool helps to locate corruption in the part.

Unigraphics cannot launch ANSYS on Windows platforms; Unigraphics can only launch ANSYS from UNIX platforms. See Section 3.2: Launching ANSYS Connection from Unigraphics (UNIX Only) for more information.



# Index

## Symbols

.sat files, 2–8

## A

ACIS geometry file, 2–8

ACIS-based CAD files

importing, 2–8

ANFO command, B–2

ANSCon Config option, 3–2

ANSCON\_CONFIG\_DIR, 2–12

ANSCON\_CONFIG\_DIR environment variable, 3–6

ANSYS

configuration settings, 3–4

launching from Pro/ENGINEER, 3–2

launching from Unigraphics, 3–5

quitting, 3–2, 3–5

starting from Unigraphics, 4–1

ANSYS Connection, 1–1

supported systems for, 1–2

ANSYS Geom option, 3–2

ANSYS\_GRAPHIC\_KEYWORD environment variable, 3–7

ANSYS\_PROE\_CMD environment variable, B–3

.asm files, 2–7

## C

CAD packages, 1–1

CAD parts, 1–1

CATIA, 2–1

CATIA version 5, 2–2

CIF, 2–4

importing, 1–3, 4–1

Parasolid, 2–5

Pro/ENGINEER, 2–7

Caltech Intermediate Format files

importing, 2–4

~CAT5IN command

, 2–2

CATIA

importing parts created in, 2–1

~CATIAIN command, 2–1

CIF files

importing, 2–4

~CIFIN command, 2–4

combining .anf files, 1–2

config.anscon file, 3–4

Configuration Editor, 3–2

configuration files

troubleshooting, B–2

configuration settings

ignored, 3–4

modifying from Pro/ENGINEER, 3–3

modifying from Unigraphics, 3–6

updating from Unigraphics, 3–7

configure option, 3–5

connection

troubleshooting, B–1

converting a CAD part, 3–2

## D

database file

clearing, 4–1

defeaturing, 4–1

## E

editing configuration settings, 3–7

entities

Parasolid, 2–5

environment variables

ANSCON\_CONFIG\_DIR, 2–12, 3–6

modifying from Unigraphics, 3–7  
, 3–7

set by ANSYS, 3–1

UGII\_BASE\_DIR, 2–12

UGII\_OPTION, 2–12, B–5

UGII\_VENDOR\_DIR, 2–12

## F

files

.cif, 2–4

combining .anf, 1–2

config.anscon, 3–4

.dlv, 2–1

.model, 2–1

protk.dat, 3–1, 3–2

.prt, 2–7, 2–10

.CATPart, 2–2, 2–8

## G

Geometry Interface

see ANSYS Connection, 1–1

## H

hardware requirements, 1–2

## I

import commands

~CAT5IN, 2–2

~CATIAIN, 2–1

~CIFIN, 2–4

~PARAIN, 2–5, 2–5

- ~PROEIN, 2–7
- ~SATIN, 2–8
- ~UGIN, 2–10
- importing, 1–1
  - .sat files, 2–8
  - ACIS-based CAD files, 2–8
  - CAD parts, 1–3
  - CATIA parts, 2–1
  - CATIA version 5 parts, 2–2
  - .cif files, 2–4
  - CIF parts, 2–4
  - Parasolid assemblies, 2–6
  - Parasolid parts, 2–5
  - Pro/ENGINEER parts, 2–7
  - Unigraphics parts, 2–10, 4–1
- importing parts
  - Parasolid, 2–5
  - Pro/ENGINEER, 2–7
- /INPUT command, 1–2
- installation prerequisites, 1–2

## L

- launch troubleshooting, B–1
- license troubleshooting, B–1

## M

- measurement units in Parasolid files, 2–6
- multiple .anf files, 1–2

## P

- ~PARAIN command
  - , 2–5
- Parasolid
  - importing parts, 2–5
  - troubleshooting file import, B–3
- Parasolid files
  - importing, 2–5
  - scaling models, 2–6
- parts
  - CATIA, 2–1
  - CATIA Version 5, 2–2
  - importing, 1–3
  - importing Parasolid, 2–5
  - opening in Unigraphics, 3–5
  - plotting volumes of, 3–2, 3–5
  - SAT, 2–8
  - troubleshooting import, B–2
  - Unigraphics, 2–10
- path environment variable, B–3
- plotting volumes in ANSYS, 3–2, 3–5
- prerequisites for installation, 1–2
- Pro/ENGINEER

- importing models from, 2–7
- launching ANSYS from, 3–2
- modifying ANSYS settings from, 3–3, 3–4
- setting command name for, B–3
- ~PROEIN command, 2–7
- protk.dat file, 3–1, 3–2
- .prt files, 2–7
  - Unigraphics, 2–10
- P\_SCHEMA environment variable, B–3

## S

- SAT files
  - troubleshooting import, B–4
- ~SATIN command, 2–8
- scaling models
  - in Parasolid imports, 2–6
- shared libraries
  - troubleshooting, B–1
- Start Ansys option, 3–5
- starting ANSYS
  - from Unigraphics, 4–1
- system requirements, 1–2
- systems, 1–2
  - supported for ANSYS Connection, 1–2

## T

- troubleshooting, B–1
  - configuration files, B–2
  - licenses, B–1
  - Parasolid file import, B–3
  - part import, B–2
  - SAT file import, B–4
  - Unigraphics file import, B–5

## U

- UGII\_BASE\_DIR, 2–12
- UGII\_OPTION, 2–12, B–5
- UGII\_VENDOR\_DIR, 2–12
- ~UGIN, 2–10
- Unigraphics
  - files created during file import, 2–11
  - importing files from, 2–10
  - launching ANSYS from, 3–5
  - modifying ANSYS configuration settings from, 3–6
  - troubleshooting file import, B–5